ANALISYS AND OPTIMIZATION OF A T-SHAPED HYDROFORMING DIE BASED ON ITS CONNECTION RADIUS

Vaceanu Bogdan Constantin[1], Gheorghe Nagit[2], Huian Vasile[3]

[1]*Gheorghe Asachi Technical University of Iasi, Romania, vaceanubogdan@tuiasi.ro.*
[2]*Gheorghe Asachi Technical University of Iasi, Romania, nagit@tcm.tuiasi.ro.*
[3]*Gheorghe Asachi Technical University of Iasi, Romania, giorden84@yahoo.com.*

Abstract: *The connection radius represents one of the main issues in the designing process of a hydroformation T-shaped die. During hydroformation the blank which is a hollow pipe deforms depending on the hydroformation die's shape. The paper presents simulations of the hydroformation process of a pipe for three connection radiuses in such way that it would highlight the advantages brought by an optimum value. The originality of this paper study consists in an optimization method based on extensive FEA analyses which results in significant time and costs savings.*

Keywords: *T-shaped hydroformation die, connection radius, FEA*

1. Introduction

Pipe hydroformation is a complex manufacturing process that is being used in the automotive industry, aviation as well as many other industries. It depends upon special working conditions but most of all on skilled specialists [1].

The hydroformation of a pipe in a T-shape has to be carried out taking into account some few conditions such as [2]:
- Die's connection radius,
- Die's inner surface quality,
- Blank's lubrication,
- Blank's choice for the material and its characteristics (physicals, chemicals and mechanicals),
- Axial punches working speed,
- Counterpunch's working pressure,
- The internal pressure needed for the hydroformation of the blank.

2. Description of the experiments

This paper presents the hydroformation process simulation in *Ansys* of a pipe in a T-shaped die. The simulations were carried out for three distinct values for the connection radius of the hydroformation die [3].
- The die with the 1 mm connection radius,
- The die with the 4 mm connection radius,
- The die with the 7 mm connection radius.

2.1 Simulation process description

The experimental mock-up consisted out of a 3D model development, the definition of the blank's material with its proprieties in *Ansys* database, the definition of constraints and working parameters such as internal pressure, the pressure applied to the axial punches and the counterpunch, or the friction coefficient between the die and the blank. The output was set to show blank's total deformation, equivalent elastic strain and equivalent *von-Mises* stress after the hydroformation took place.

2.2 Experimental model mock-up

In order to reduce the working times as well as to ease the computational effort, a 3D simplified model was developed. It respects both the dimensions and the characteristics of the real physical assembly as well as the experimental stand and it allows definition of the same hydroformation actions, parameters and features.

The simplified assembly consists of two main components: the die and the blank. The real assembly (Fig. 1) consists of many more components and the definition of those in the hydroformation process would have had a significant impact on the computational level allowing fewer simulations related cycles because of its complexity degree.

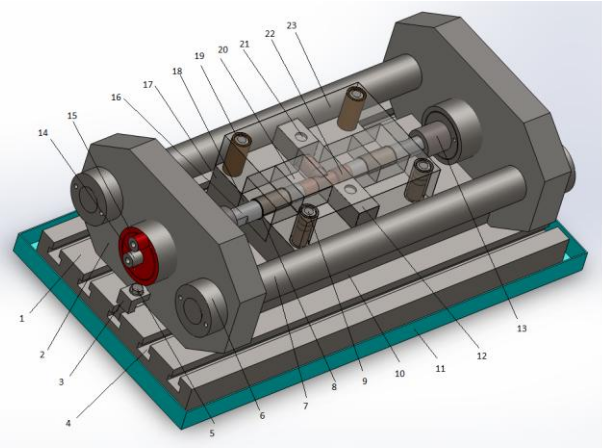

Figure 1: *Schematics of the experimental hydroformation stand*.

Fig. 1 shows all the components of the experimental stand as follows: *1-Base plate, 2-Sideway plate, 3-Fastening element, 4-T-shaped guide, 5-Screw, 6-Puller nut, 7-Puller, 8-Roller bearing, 9-Guidance element, 10-Spacer, 11-Tray, 12-Retention block, 13-Hydraulic cylinder, 14-Cylinder joint, 15-Cover, 16-Axial punch, 17-Lower die, 18-Guidance part, 19-Guidance bush, 20-Sideway element, 21-Retainer cylinder, 22-Hydroformed pipe* and *23-Upper die*. The 3D complete model of the experimental stand shown above was designed in Solid Works 2012 CAD environment. The components were individually drafted and then assembled in correspondence with the values provided by extensive measurements carried out on each component. The 3D assembled model was saved initially in the *Assembly* format for it to be accessible for any modification that might intervene and then in a *Parasolid* format which can be read by *Ansys*.

In order to simulate the hydroformation process a simplified version of the experimental stand was used (Fig. 2). The mock-up consists of two components which have the same geometrical features of the die assembly and the blank to be deformed.

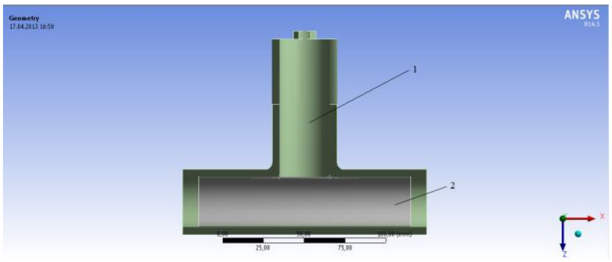

Figure 2: *Simplified mock-up of the experimental stand.*
1 - The hydroformation die,
2 - The blank.

The others components of the real assembly are no longer necessary as they would have no influence what so ever on the simulation process in *Ansys*. All constraints, parameters and output are available by means of software.

2.3 Material definition in *Ansys*

The first step in the simulation of the hydroformation process is to properly define the blank's material in order to be fully analyzed and "*understood*" by the software. Its proprieties are very important since they will have a direct influence on how will the material behave under given constraints and working conditions, especially on how will it flow taking the form imposed by the die's T-shape. *Ansys* comes with its own database with various materials which can be assigned to a part or assembly. If the material is not in its database, it can be defined and then saved by adding its proprieties and even name it accordingly. The definition also allows various behavior laws to be attached in such way that the saved material would behave exactly like the real one in the same working conditions.

Table 1 presents the proprieties of OLC15 blank's material which was used in the experiments.

Table 1: *OLC15 proprieties*.

Chemical composition	**Percentage**
As	0.050 %
C	0.120 – 0.180 %
Cr	0.300 %
Cu	0.300 %
Mn	0.350 – 0.650 %
Ni	0.300 %
P	0.040 %
S	0.045 %
Si	0.170 – 0.370 %
Mechanical proprieties	**Units**
Yield stress	350 MPa
Tensile strength	590 – 780 MPa
Elongation	14 %
Physical proprieties	**Units**
Density	7.86 kg/m^3
Young modulus	203 GPa
Poisson ratio	0.27 – 0.30

The paper presents the results for just laminated quality steel but the study was performed considering other types of materials as well. All this proprieties were input in *Ansys* database under the name of OLC15.

2.4 Geometry setup

Ansys allows the design of 3D models in one of its modules but it lacks several modeling features and that made it unattractive for the 3D model design development. Thus the authors turned to a more advanced software solution such as Solid Works 2012 since *Ansys* reads most of CAD related files (*CATProduct, CATPart, IGS, STEP, Parasolid, etc.*). All of the experimental stand's components were designed and assembled in full correspondence with the dimensions of the physical analyzed parts. The model was build out of each designed part and then saved as an assembly for it to be modifiable if necessary or any adjustments required. Then the same assembly was saved as a *Parasolid* in order to be accessible to *Ansys*. The model was significantly simplified taking out any unnecessary elements that would have aggravated the computational process. All geometrical features are fully read and a valid mesh can be developed based on the imported 3D model.

2.5 Connections management

The second step in the process of any simulation consists of assembly behavior setup. In fact, each component has to be constrained in such manner that it would simulate the real working environment. *Ansys* allows the user to define each component as a rigid or a flexible body. This has a direct influence on the body because it determines the outcome of the analysis. The output, strain stress and deformation should only be set to the deformed analyzed body and not to the entire assembly. That being said, if the results are set to the entire assembly, the displayed values would include also the die which is not being affected by the hydroforming process. Thus, only the pipe is set up to display the results of hydroformation upon it. More, the software allows the user to introduce probes (*sensors*) which can give detailed information about a local area from the analyzed part. However this option is not suited for the present analysis since the pipe is a hollow cylinder and has no sharp edges, angles or is formed out of multiple free surfaces with different degrees of complexity from the geometrical point of view.

The parts that are being set to rigid are not affected during analyses since they don't get deformed. In the case of the pipe which has been set to flexible, the software pushes the process depending on how much it is needed. The deformation can be set up to the breaking point depending on the material's proprieties, loads, constraints and the number of cycles. Of course, the process can be pushed up even further to show what happens beyond the yield limit and how material cracks and where and why.

As we have highlighted previously, the behavior of each component has to be set. *Ansys* allows the connection between elements to be defined as desired: bonded, with or without friction, rough or even spot welded and how they interact with each other: freely, with or without contact. In this study the authors optioned for contact between surfaces with friction. The friction can be set up between two or more surfaces. The friction coefficient was set up at 0.1 between the exterior cylindrical surface of the pipe and the inner shell of the die.

2.6 Meshing in *Ansys*

In order to perform any type of FEA a meshing process of a part or assembly has to be performed. It will divide and subdivide areas, regions or elements which are interconnected into a mesh connected by vertexes or nodes. *Ansys* can automate this process offering the possibility to choose between several options of meshing such as *Tetrahedrons*, *Hex Dominant*, *Sweep* or *Multi Zone*. Each of these options is best suited in different types of FEA analyses. After many tryouts the authors chose for the meshing of the pipe the *Tetrahedrons* method with the *Patch Conforming* option. The die was also individually meshed with a *Body Sizing* option which allowed a 10 mm vertex size. The vertex size for the blank was set to 7 mm (Fig. 3). The next step consisted of choosing a type of mesh for special areas that are of interest in the analysis (Fig. 3).

For both the pipe's outer cylindrical surface and the die's T-shaped inner shell surface the authors set a 2 mm vertex size with a *Face Sizing* option.

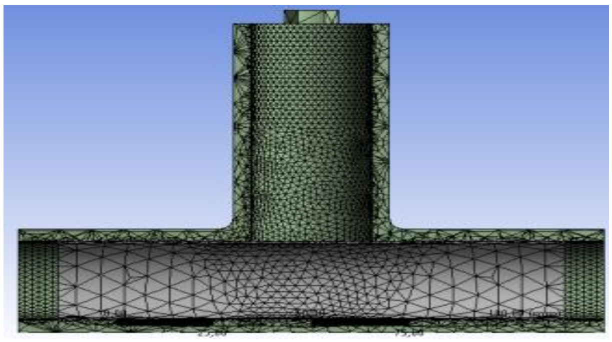

Figure 3: *The meshing process of the assembly.*

The same value and option were set for the connection radiuses were the T is formed. The inner cylindrical surface of the blank was also meshed with a *Face Sizing* option with a 1.5 mm vertex size. This allows the software to "*understand*" that those are the targeted areas which are subjected to deformation and thus the results will be more accurately. Meshing separately components allowed the authors to obtain far more sensitive data especially in the areas where the pipe gets T-formed.

Ansys shows 82225 vertexes and 44652 elements for the blank's inner and outer surfaces after the meshing process.

2.7 Constraints and loads definition

Fig. 4 shows all the constraints and loads that were defined for the hydroformation simulation process: *A-Fixed support (the die), B-Internal pressure (applied to the inner cylindrical surface of the blank), C, D- Displacements (applied to the axial punches)*. The die was set to fix in order to deny all degrees of freedom of movement or deformation. We remember that all components, except the blank were set to rigid and that results were asked only for the pipe after the hydroformation was simulated. *Ansys* goes further and allows the user to gradually input the load.

The authors have indicated a building pressure up to 40 MPa in 20 steps from scratch inside blank B as we can see from Fig. 4. Each step or cycle will add a 2 MPa pressure load to the process. The load is applied on the whole surface in such way that there are not any weak points on the surface which will allow different values of the same load.

The counterpunch is disregarded and the blank deforms freely under the combined action of the two axial punches which are simulated to displace by 30 mm each. Each step will add a further 1.5 mm per second displacement in order to deform the analyzed component.

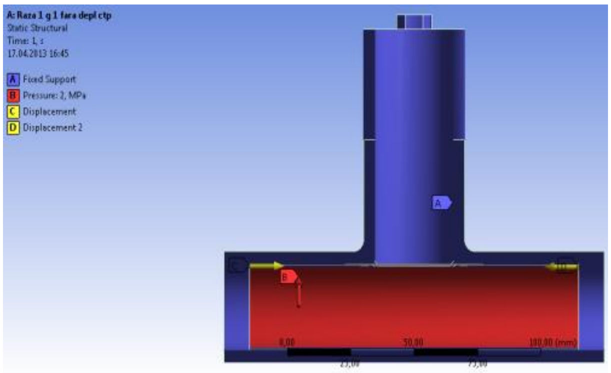

Figure 4: *Constraints and loads definition*.

As we mentioned above there is no need for all components of the experimental stand, namely the axial punches or the counterpunch.

The simulations were performed for three different experimental models as follows:
- M1: *connection radiuses were the T forms are of 1 mm in size*,
- M2: *connection radiuses were the T forms are of 4 mm in size*,
- M3: *connection radiuses were the T forms are of 7 mm in size*.

2.8 Numerical results

After the simulations were carried out, the authors could evaluate the behavior of cylindrical blanks (*pipes*) made out of a common used type of steel, during and after hydroformation process in a T-shaped die. The results, as expected are different for each considered model highlighting the importance of an optimal value for the connection radius that determines the T shape of the analyzed component.

Table 2: Numerical r*esults for the considered models.*

Model no.	**Total deformation [mm]**	**Eq. Elastic Strain [mm/mm]**	**Eq. *von-Mises* Stress [MPa]**
M1	27	0.0025704	517.23
M2	27	0.0029253	590.23
M3	27	0.0029388	592.82

The deformations displayed for each case in Table 2 are for a 130 mm in length and 1 mm in thickness type of blank.

3. Interpretation of results

3.1 Model 1

Fig. 5 shows the resulted total deformation of model 1 after the hydroformation process.

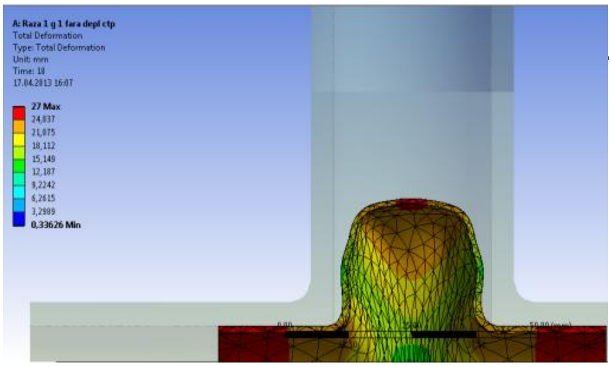

Figure 5: *Total deformation for model 1*.

For a 1 mm in value for the connection radius of the T-shaped die the software shows 0.0025704 mm/mm for the resulted equivalent elastic strain.

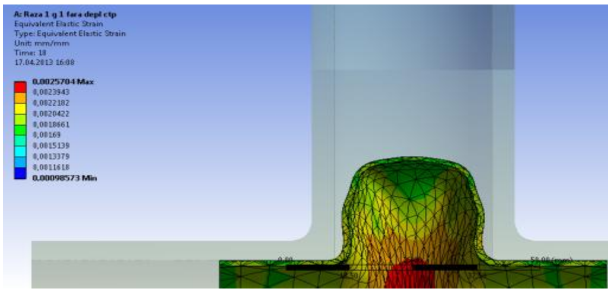

Figure 6: *Equiv. elastic strain for model 1*.

As expected, as the material is pulled into the T form the strain gets higher as it is marked in red color. The strain is more evident in the yield areas as the material starts to flow.

Fig. 7 shows the resulted equivalent *von-Mises* stress of model 1 after the hydroformation process.

For a 1 mm in value for the connection radius of the T-shaped die the software shows 517.23 MPa for the resulted equivalent *von-Mises* stress. As the pressure builds up the blank starts to deform, but due to the small value for the connection radius is unable do so accurately. We

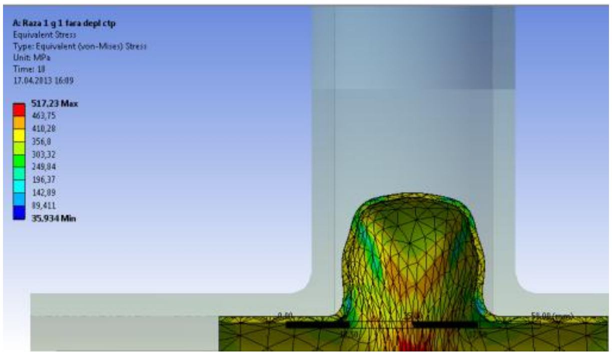

Figure 7: *Equiv. von-Mises stress for model 1*.

can see that the blank has room to further deform but it won't without a bigger pressure environment.

3.2 Model 2

Fig. 8 shows the resulted total deformation of model 2 after the hydroformation process.

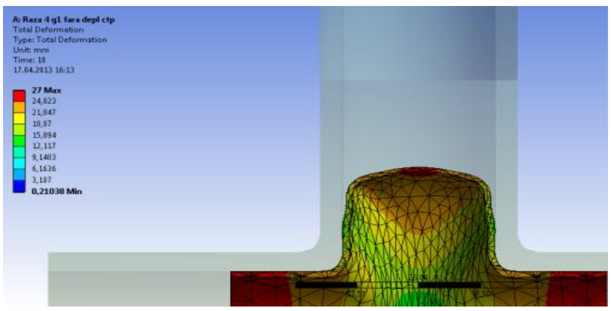

Figure 8: *Total deformation for model 2*.

For a 4 mm in value for the connection radius of the T-shaped die the software shows the same value as in case of model 1 of 27 mm for the resulted total deformation.

Fig. 9 shows the resulted equivalent elastic strain of model 2 after the hydroformation process.

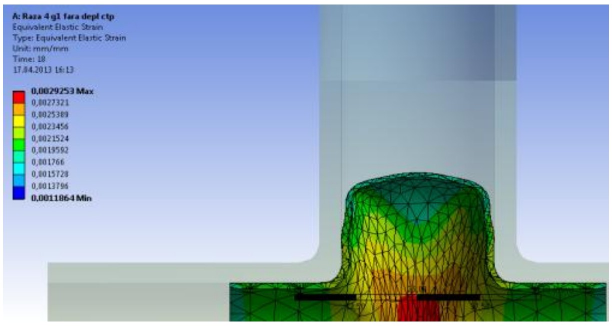

Figure 9: *Equiv. elastic strain for model 2*.

For a 4 mm in value for the connection radius of the T-shaped die the software shows 0.0029253 mm/mm for the resulted equivalent elastic strain.

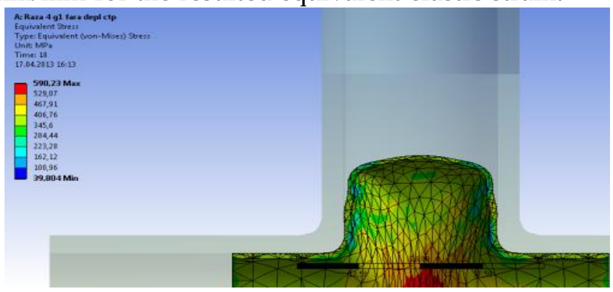

Figure 10: *Equiv. von-Mises stress for model 2*.

Fig. 10 shows the resulted equivalent *von-Mises* stress of model 2 after the hydroformation process.

For a 4 mm in value for the connection radius of the T-shaped die the software shows 590.23 MPa for the resulted equivalent *von-Mises* stress. For this model we have noticed a 73 MPa increased stress from the first one. The blank has deformed differently and we see that the bulged area is now bigger.

3.3 Model 3

Fig. 11 shows the resulted total deformation of model 3 after the hydroformation process.

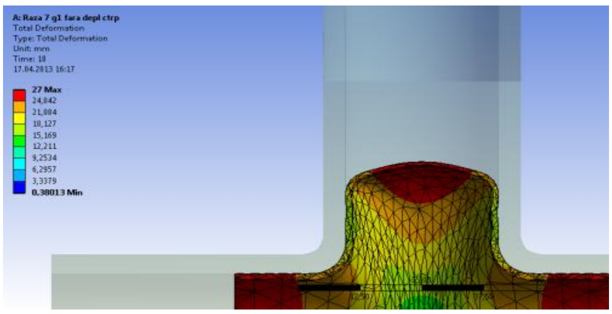

Figure 11: *Total deformation for model 3*.

For a 7 mm in value for the connection radius of the T-shaped die the software shows the same value of 27 mm for the resulted total deformation.

Fig. 12 shows the resulted equivalent elastic strain of model 3 after the hydroformation process.

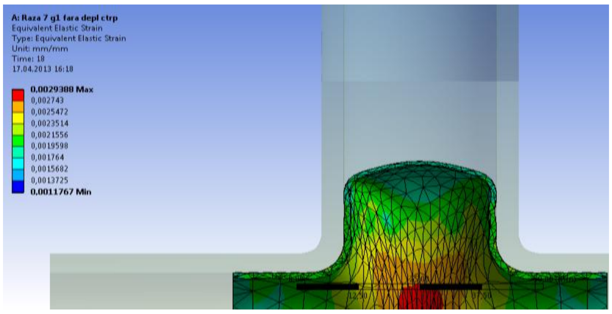

Figure 12: *Equiv. elastic strain for model 3.*

For a 7 mm in value for the connection radius of the T-shaped die the software shows 0.0029388 mm/mm for the resulted equivalent elastic strain.

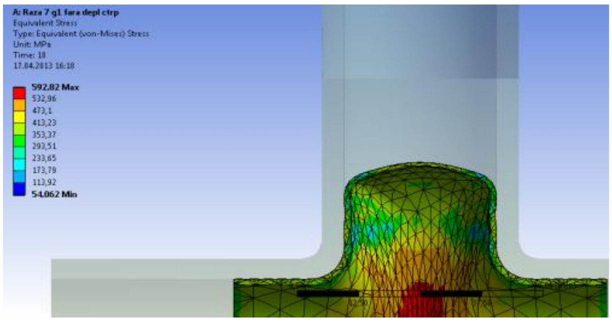

Figure 13: *Equiv. von-Mises stress for model 3.*

Fig. 13 shows the resulted equivalent *von-Mises* stress of model 3 after the hydroformation process.

For a 7 mm in value for the connection radius of the T-shaped die the software shows 592.82 MPa for the resulted equivalent *von-Mises* stress.

4. Conclusions

Analyses and simulations were carried out in case of a pipe's hydroformation process which had determined a value for the connection radius that corresponds to an optimized inner shell of a T-shaped hydroformation die.

Simulations showed that even if the resulted total deformation had the same value, some of the deformed area was different increasingly bulging as the radius grew bigger. This means that the yield limit of the material given by Young's modulus is reached as the material is allowed to flow by means of a bigger connection radius. If the differences between the first two models in terms of *von-Mises* stress are significant, the biggest radius brings only a slight increase of this stress value as the material settles in the die's inner shell.

As result by trial & errors the authors came to the conclusion that the connection radius of a T-shaped die for this case study best value is of 7 mm and that beyond this value the blank will crack as tests on the real experimental stand showed.

The originality of this paper study consists in an optimization method based on extensive FEA analyses which results in significant time and costs savings, because depending on the die's dimensions an estimation can be made upon the type of parts that may be hydroformed.

5. Acknowledgement

This paper was realized with the support of POSDRU CUANTUMDOC "*Doctoral studies for European performances in research and innovation*" ID79407 project funded by the European Social Fund and Romanian government.

References

[1] Bing L., Nye T. J., Metzger D., *Multi-objective optimization of forming parameters for tube hydroforming process based on the Taguchi method*, The International Journal of Advanced Manufacturing Technology, 2006, available at www.springerlink.com.

[2] Ceretti E., Braga D., Giardini C., *Optimization of process parameters and part geometry for high diameter tube hydroforming*, International Journal of Material Forming, 2010, available at www.springerlink.com.

[3] Kadkhodayan M., Moghadam A. E., *Optimization of load paths in X- and Y-shaped hydroforming*, International Journal of Material Forming, 2011, available at http://link.springer.com.